ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

Computational Estimation of Flow through the Convergent Divergent Nozzle

D. Chaitan Kumar
M.Tech (Thermal Engineering) Student
Department of Mechanical Engineering
Aditya Institute of Technology and Management, Tekkali.

Dr. N. Haribabu
Professor
Department of Mechanical Engineering
Aditya Institute of Technology and Management, Tekkali.

ABSTARCT

In this thesis, CFD analysis of flow within, Convergent – Divergent rectangular supersonic nozzle and supersonic impulse turbine with partial admission have been performed. The analysis has been performed according to shape of a supersonic nozzle and length of axial clearance, objective is to investigate the effect of nozzle-rotor interaction on turbine's performance.

It is found that nozzle-rotor interaction losses are largely dependent on axial clearance, which affects the flow with in nozzle and the extent of flow expansion. Therefore selecting appropriate length of axial clearance can decrease nozzle-rotor interaction losses. The work is carried in two stages:
1. Modeling and analysis of flow for rectangular convergent divergent supersonic nozzle.
2. Prediction of optimal axial gap between the Nozzle and rotor blades by allowing the above nozzle flow.
In the present work, flow through the convergent divergent nozzle study is carried out by using a finite volume commercial code, FLUENT 6.2. The nozzle geometry is modeled and grid is generated using GAMBIT 2.3 Software. Computational results are in good agreement with the experimental ones.

1.1 INTRODUCTION:

Advances in computing technology, software and hardware have revolutionized the design process of engineering vehicles such as aircrafts, automobiles and ships. Many commercial software packages are being used in the design as well as analysis processes which not only save lead time and costs of new designs, but also are used to study systems where controlled experiments are difficult or impossible to perform. In the area of fluid dynamics, there are many commercial computational fluid dynamics (CFD) packages available for modeling flow in or around objects.

Computational fluid dynamics (CFD) has been constantly developed over the past few decades and now both commercial and research codes can provide more and more robust and accurate results. Combined with the use of wind tunnel test data, CFD can be used in the design process to drive geometry change instead of being used mainly as a design validation tool. Computational Fluid Dynamics (CFD) has become an integral part of the engineering design and analysis environment of many companies that require the ability to predict the performance of new designs or processes before they are ever manufactured or implemented. One of the most critical requirements for any CFD tool used for thermal applications is the ability to simulate flows along nozzles, turbines. Such challenging features as pressure gradients, shocks, velocity distribution, eddy location, stream line curvature, and stream wise vortices pose a challenge for computation. The small margins of improvement that are usually targeted in nozzle and turbines design today require precise tools capable of discerning small differences between alternative designs. Custom modeling tools that are based as simplified numerical methods and assumptions cannot provide the accuracy that can be obtained with CFD, which offers mainly inherent advantages for ex: it offers quick and cheap solutions in comparison to experimental solutions and more accurate in comparison to empirical methods used in design. Accurate simulation of flows through the nozzle is important for prediction of velocity pattern and pressure patterns.

The current study aims analysis of flow through the nozzle and prediction of optimal axial clearance. Solution of flow along the nozzle involve only one phase of gas. Results are verified with the experimental data. As a part of project work nozzle study is carried out and with using same nozzle, axial gap / (clearance determination) is analyzed. The results are in good agreement with the experimental ones.

1.2 OBJECTIVES:

The objective of present work is
1. Modeling and meshing of nozzle geometry.
2. Validate the CFD results of nozzle flow with theory and experiments.
3. Modeling and meshing of nozzle and turbine blades as a case of partial admission type.
4. Validate the CFD results of nozzle and turbine blades with experimental data.

This aims to predict the following:

- Estimation of velocity at nozzle exit as a case of whether supersonic (or) not.
- Estimation of nozzle and turbine rotor gap under static condition of rotor.
- Flow visualization.

1.3 HISTORY:

The distinctive feature of our civilization, one that marks it off from all others, is the wide use of mechanical power. At one time the primary source of power for the work of peace and war was chiefly man’s muscles, even after animals had been trained to help, and after the wind and running streams had been harnessed. But the great step was taken when man learned to convert the heat of chemical reactions into mechanical energy. Machines which serve this purpose are known as heat engines. The gas turbine in its most common form is a heat engine operating by means of a series of processes consisting of compression of air taken from the atmosphere, increase of gas temperature by constant-pressure combustion of fuel in the air, expansion of hot gases and, finally, discharge of the gases to atmosphere, the whole process being a continuous flow process. It is thus similar to the S.I. and C.I. engines into working medium and, internal combustion but is akin to the steam turbine in its aspect of the steady flow of the working medium. Of the early inventors recognizing the possibilities of the gas turbine the most outstanding was John Barber, an Englishman, whose patent specification of 1791 is of special significance in being the first recorded description of the gas turbine, and also in anticipating the "constant pressure" method which has since been adopted in the most recent successful turbines.

The next important step in the development of the gas turbine appears to have been due to Rene Armengaud and Charles Lemal, two French inventors who built an engine in 1894 which was claimed to have worked satisfactorily over a period of some years. However, there was a lapse of many years, until in 1939, a Brown Boveri unit for emergency electric power supply was put into operation in Neuchatel, Switzerland, the output being 4000Kw and the efficiency 18%. There are two major reasons why the application of internal combustion to turbine machinery waited so long for exploitation. The first is that the working medium has to be compressed to the maximum pressure level of the cycle as gas and, as this represents a large power input, it must be done efficiently for the cycle to be effective.

Until recent years, turbo machines for compression did not have efficiency high enough for the purpose and it required the methods and outlook of the science of aerodynamics to come to fruition before this was accomplished. The second reason is that, given a reasonable compression and expansion efficiency, it still requires a high value of maximum cycle temperature in order to achieve a useful plant output and efficiency. The development of materials capable of long life at the temperature as required was a result of metallurgical progress in the 1930's of the manufacture of alloy steel for the values of reciprocating engines for aircraft. The latter represented a firm market, whereas the gas turbine was only a dubious one, for which the expenditure on research was not justified.

Once metals for such temperature levels were available, the gas turbine became a practical possibility. During the period between 1904 and 1933, the Holzwarth constant volume gas turbine was developed and several units were put into service. Since the constant-volume combustion type turbine has not prospered to-date in comparison with the constant-pressure combustion type, it is not discussed further. During this period, the development of the supercharger for the reciprocating aircraft engine was a great stimulus to the gas turbine. Both in the gear-driven form and in the turbo-supercharger type, it led to advances in the centrifugal compressor and, in the turbo-supercharger form, it also demonstrated that a turbine operating at a high peak temperature was possible. The development of the turbo supercharger is associated with the names of Moss in America .and Buchi in Switzerland.

In 1930's considerable attention was given to the jet propulsion of aircraft, and some of the proposed schemes utilized the gas turbine as the source of high-velocity gas. The pioneers of to-day's turbo jet engine is undoubtedly Whittle, whose British patent of 1930 embodies both axial flow and centrifugal compressors, constant-pressure combustion, axial flow turbine and propulsion nozzle. In 1935, von Ohairi in Germany patented a unit with centrifugal compressor and radial-outflow turbine. It was the military application of the turbo-jet during World War II which gave such impetus to the subsequent development of the gas turbine in the post-war years. The research efforts sponsored by governmental agencies resulted in tremendous development of the special metals needed for continuous high-temperature service at a high stress level. In addition, it provided a core of Engineers and technicians trained in the aircraft development, who to a considerable extent spread out the design knowledge to industrial applications

Furthermore, although the transition from aircraft use to industrial use is not direct the obvious success of gas turbine in the air gave acceptance to its application on ground. To-day the gas turbine is pre-eminent. as an aircraft power plant, with outputs ranging from a few hundreds of kilograms of thrust to over 10,000 kilograms. As a shaft power unit, one of the smallest in regular service is about 5 h.p., while at the other end of the scale is a unit of over 35000 h.p

2. FLUID GOVERNING EQUATIONS:

The governing equation of fluid represents mathematical statement of the conservation laws of physics. The individual differential equations that we shall encounter express a certain conservation principle. Each equation employs a certain physical quantity and its dependent variable and implies that there must be a balance among the various factors that influence variables. The dependent variables of these differential equations are usually specific properties. The terms in the differential equation of this type denotes influence on a unit volume basis. The fundamental equations of fluid dynamics are based on the following universal law of conversation. They are:

1) Conservation of mass.
2) Conservation of momentum.
3) Conservation of energy

2.1 CONTINUITY EQUATION:

The equation based on the principle of conservation of mass is called continuity equation. The conservation of mass law applied to a fluid passing through an infinitesimal, fixed control volume yields the following equation of continuity,

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}+\frac{\partial(\rho w)}{\partial z}=0$$

$$\frac{\partial \rho}{\partial t}+\nabla.(\rho V)=0$$

Where ‘ρ’ is the density, ‘V’ is the fluid velocity .For an incompressible flow, The density of each fluid element remains constant.

2.2 MOMENTUM EQUATION:

it’s based on the law of conservation of momentum or on the principle of momentum, Which states that, the net force acting on fluid mass is equal to the change in momentum of flow per unit time in that direction. The Navier Stokes equation in conservative form

$$\frac{\partial(\rho u)}{\partial t}+\frac{\partial(\rho u^2)}{\partial x}+\frac{\partial(\rho uv)}{\partial y}+\frac{\partial(\rho uw)}{\partial z}=-\frac{\partial p}{\partial x}+\frac{\partial \tau_{xx}}{\partial x}+\frac{\partial \tau_{yx}}{\partial y}+\frac{\partial \tau_{zx}}{\partial z}+\rho f_x$$

$$\frac{\partial(\rho v)}{\partial t}+\frac{\partial(\rho uv)}{\partial x}+\frac{\partial(\rho v^2)}{\partial y}+\frac{\partial(\rho vw)}{\partial z}=-\frac{\partial p}{\partial y}+\frac{\partial \tau_{xy}}{\partial x}+\frac{\partial \tau_{yy}}{\partial y}+\frac{\partial \tau_{zy}}{\partial z}+\rho f_y$$

$$\frac{\partial(\rho w)}{\partial t}+\frac{\partial(\rho uw)}{\partial x}+\frac{\partial(\rho vw)}{\partial y}+\frac{\partial(\rho w^2)}{\partial z}=-\frac{\partial p}{\partial z}+\frac{\partial \tau_{xz}}{\partial x}+\frac{\partial \tau_{yz}}{\partial y}+\frac{\partial \tau_{zz}}{\partial z}+\rho f_z$$

Unsteady *Convective* *Pressure* *Diffusive* *Source*

The Navier Stokes equation forms the basis of any viscous flow solutions. The main thrust of present day research in CFD and heat transfer in turbulent flow is through the time averaged Navier –Stokes equations.These equations are also referred to as the Reynolds averaged Navier-Stokes equations.

2.3. ENERGY EQUATION:

This equation is based on the principle of conservative of energy, which is generally referred to as the first law of thermodynamics, which states that “energy” Can be neither created nor destroyed but can only be changed from one form to another form.

Energy equation in conservative form

$$\begin{aligned}\frac{\partial}{\partial t}\left[\rho\left(e+\frac{V^2}{2}\right)\right]+\nabla\cdot\left[\rho\left(e+\frac{V^2}{2}\right)\mathbf{V}\right]=\rho\dot{q}+\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right)\\+\frac{\partial}{\partial z}\left(k\frac{\partial T}{\partial z}\right)-\frac{\partial(up)}{\partial x}-\frac{\partial(vp)}{\partial y}-\frac{\partial(wp)}{\partial z}+\frac{\partial(u\tau_{xx})}{\partial x}\\+\frac{\partial(u\tau_{yx})}{\partial y}+\frac{\partial(u\tau_{zx})}{\partial z}+\frac{\partial(v\tau_{xy})}{\partial x}+\frac{\partial(v\tau_{yy})}{\partial y}\\+\frac{\partial(v\tau_{zy})}{\partial z}+\frac{\partial(w\tau_{xz})}{\partial x}+\frac{\partial(w\tau_{yz})}{\partial y}+\frac{\partial(w\tau_{zz})}{\partial z}+\rho\mathbf{f}\cdot\mathbf{V}\end{aligned}$$

2.4 EQUATION OF STATE:

Equation of state is defined as type of equation, which gives the relationship between the pressure, temperature and specific volume of gas ***(P*v= MRT)**

Where P is the pressure in bar, ’v’ is the specific volume in m^3/sec, ‘M’ is the mass in ‘kg’, ‘R’ is the universal gas constant in j/kgk and ‘T’ is the temperature in Kelvin

3. FLUENT:

This chapter describes the features of the analysis software used in the present work FLUENT6.2 FLUENT is a state -of –the art computer program for modeling fluid flow and heat transfer in complex geometries. FLUENT provides complete mesh flexibility, solving your flow problems with unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2Dtriangular/quadrilateral, 3D tetrahedral/hexahedral /pyramid/wedge, and mixed (Hybrid) meshes. FLUENT also allows you to refine or coarsen your grid based on the flow solution. This solution –adaptive grid capability is particularly useful for accurately predicting flow fields in regions with large gradients, such as free shear layers and boundary layers. In comparison to solutions on structured or block grids, this feature significantly reduces the time required to generate a “good grid. Solution adaptive refinement makes it easier to perform grid refinement studies and reduces the computational effort required to achieve a desired level of accuracy, since mesh refinement is limited to those regions where greater mesh resolution is needed

3.1 FLUENT CAPABILITIES:

The **FLUENT** solver has the following modeling capabilities Flows in 2D or3D geometries using unstructured solution-adaptive triangular/tetrahedral, quadrilateral l/ hexahedral, or mixed (hybrid) grids that include prisms (wedges) or pyramids. (Both conformal and hanging-node meshes are acceptable.)

- Incompressible or compressible flows
- Steady state or transient analysis
- In viscid, laminar, and turbulent flows
- Newtonian or non Newtonian flows

ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal

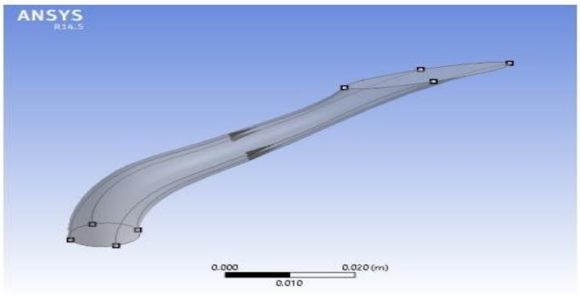
- Convective heat transfer, including natural or forced convection
- Coupled conduction/convective heat transfer
- Radiation heat transfer
- Inertial (stationery) or non inertial (rotating) reference models
- Multiples moving reference frames, including, sliding mesh interfaces and mixing planes for rotor/stator interaction modeling.

These capabilities allow fluent to be used for a wide variety of applications, including the following

- Process and process equipment applications
- Power generation and oil/gas and environmental applications
- Aerospace and turbo machinery applications
- Automobile applications
- Heat exchanger applications
- Electronics/HVAC appliances
- Materials processing applications
- Architectural design and fire research

MODELING OF THE COMPONENTS:

Modeling the Super Sonic Nozzle:



MODELING THE BLADE PROFILE

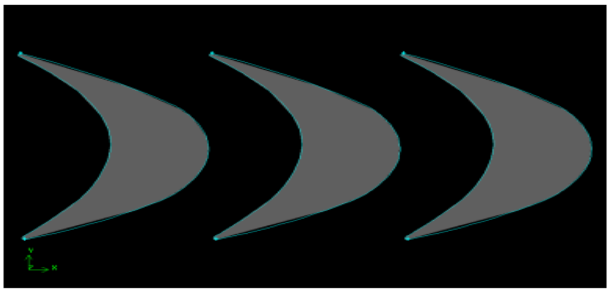


FIGURE Series of Rotor Blade

MODELING OF NOZZLE AND TURBINE BLADES AS PARTIAL ADMISSION CASE

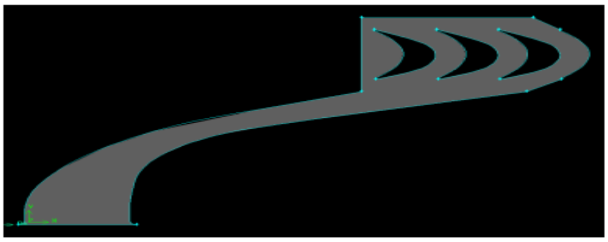


FIGURE Solid Model as a Partial Admission Case of Nozzle and Rotor Blades with 3mm axial gap.

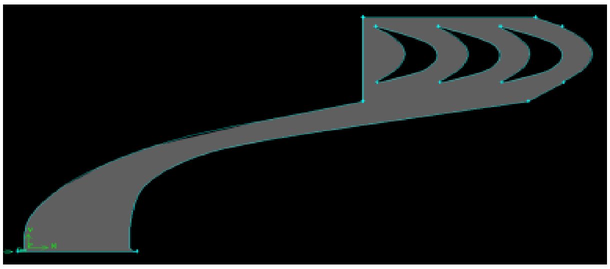


FIGURE: Solid Model as A Partial Admission Case of Nozzle andRotor blades with 4mm axial gap.

FIGURE: Solid Model as a Partial Admission Case of Nozzle and Rotor Blades with 5mm axial gap.

Mesh Generation for 2D C-D nozzle:

A 2D nozzle profile is modeled in GAMBIT. To generate the structured grid as shown in with quadrilateral elements Gambit is used. This mesh are in good agreement with turbulence model gave good results, which otherwise didn’t match with the experimental ones. Nozzle inlet as pressure inlet and outlet as pressure outlet and remaining as walls the nozzle path are defined as fluid. File is the saved for analysis in fluent.

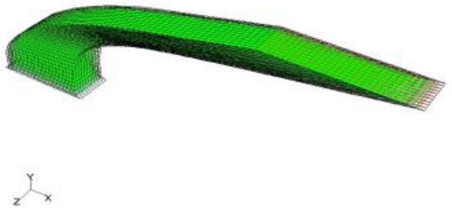


FIGURE : Meshed View of 3 D Nozzle Profile.

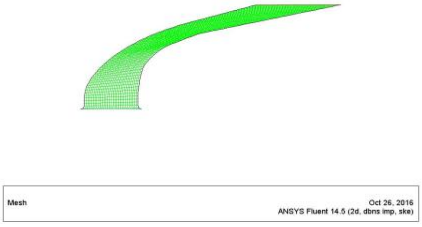


FIGURE : Meshed View of 2 D Nozzle Profile.

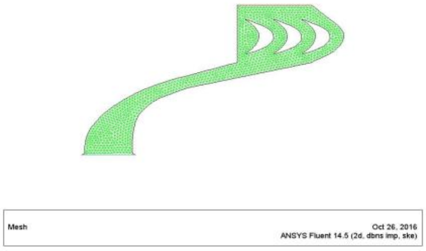


FIGURE : Meshed View as a Partial admission Case of Nozzle And rotor blades with 3mm axial gap.

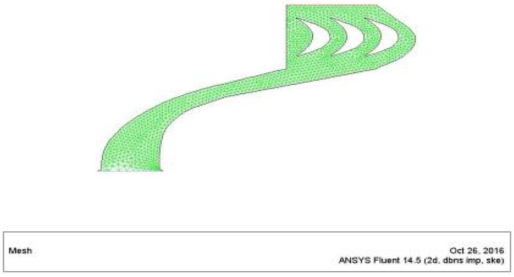


FIGURE : Meshed View as a Partial Admission Case of Nozzle and rotor blades with 4mm axial gap.

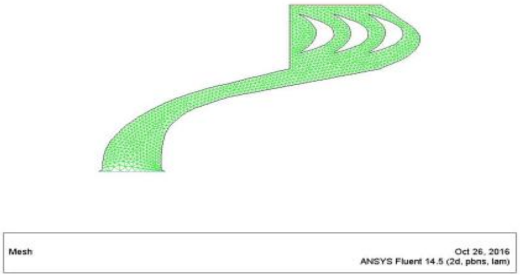


FIGURE : Meshed View as A Partial admission Case of Nozzle and rotor blades with 5mm axial gap.

7.1 ANALYSIS OF C-D RECTANGULAR NOZZLE:

The analysis is carried in fluent software by importing the meshed file saved in Gambit. The steps that are followed are given below which include all the conditions and the boundaries value for the problem statement.

7.2 CHECKING OF MESH AND SCALING:

The fluent solver is opened where 2DDP is selected and then the importing of the meshed file is done. The meshed file then undergoes a checking where number of grids are found. After this grid check is done following which smoothing and swapping of grid is done. Following this the scaling is done. Scale is scaled to mm. Grid created was changed to mm. After this defining of various parameters are done.

7.3 SOLVER AND MATERIAL SELECTION AND OPERATING CONDITION DEFINING:

The solver is defined first. Solver is taken as coupled based and formulation as implicit, space as 2D and time as steady. Velocity formulation as absolute and gradient options as green gauss cell based are taken. Energy equation is taken in to consideration. The viscous medium is also taken. They analysis is carried using K-epsilon turbulence model.

The selection of material is done. Material selected is gas. The properties of gas taken as follows: Density as ideal gas Cp (Specific heat capacity) = 2034.6J/Kg.K

Thermal Conductivity	=0.0706 W/M-K
Viscosity	=6.-7 e-5 (Kg/M-S)
Molecular weight	=23.05 (Kg/Kg-Mol)

The analysis is carried out under operating condition of zero Pascal. Gravity is not taken in to consideration.

7.4 BOUNDARY CONDITIONS:

7.4.1 Nozzle Inlet:

Pressure inlet was taken as inlet for nozzle the value of pressure is 8101325 Pascal. Initial gauge pressure was taken as 7898681 Pascal. Temperature was taken as 1583K.

7.4.2 Nozzle Outlet:

The nozzle outlet is set as pressure outlet with a value of 13e5.

7.4.3 Controls set up:

The solution controls are set as listed below: The under relaxation factor was set as given.
Turbulence Kinetic Energy 0.8

Turbulence Dissipation rate	0.8
Turbulence Viscocity	1

7.4.4 Discretization Equation is selected as given:

Flow (Second order up wind) Turbulence Kinetic Energy (1st order upwind) Turbulence dissipations rate (1st order upwind)

7.4.5 Initialization:

Solution initialization is done. Initial values of velocity are taken as 186.3 m/s for y direction. Temperature is taken as 1583K Residual monitoring is done and convergence criteria are set up. The convergence criteria of various parameters are listed below.

Continuity -	0.001
X Velocity-	0.001
Y Velocity-	0.001
Energy -	0.001

The number of iterations is set up and iterations starts.
The iteration continues till the convergence is reached and convergence history

7.5 ANALYSIS OF NOZZLE AND TURBINE ROTOR BLADES AS A CASE OF PARTIAL ADMISSION:

The analysis is carried in fluent by importing the meshed file saved in Gambit. The steps that are followed are given below which include all the conditions and the boundaries value for the problem statement, for varied axial gaps of nozzle and turbine rotor blades as a case of partial admission.

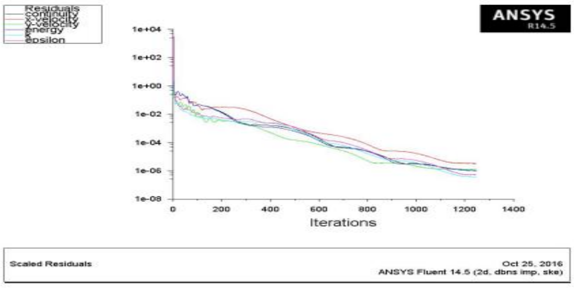


FIGURE Convergence History For C-D Nozzle.

7.5.1 Checking of Mesh and Scaling:

The fluent solver is opened where 2DDP is selected and then the importing of the meshed file is done. The meshed file then undergoes a checking where number of grids are found. After this grid check is done following which smoothing and swapping of grid is done. Following this the scaling is done. Scale is scaled to mm. Grid created was changed to mm. After this defining of various parameters are done.

7.5.2 Solver and Material Selection and operating condition defining:

The solver is defined first. Solver is taken as Segregated based and formulation asimplicit, space as 2D and time as steady. Velocity formulation as absolute and gradient options as green gauss cell based are taken. Energy equation is taken in to consideration. The viscous medium is also taken. They analysis is carried using K-epsilon turbulence model. The selection of material is done. Material selected is gas.

The properties of gas taken as follows:

Density as ideal gas Cp (Specific heat capacity)	=2034.6J/Kg.K
Thermal Conductivity	=0.0706 W/M-K
Viscosity	=6.-7 e-5 (Kg/M-S)
Molecular weight	=23.05 (Kg/Kg-Mol)

The analysis is carried out under operating condition of zero pascal. Gravity is not taken in to consideration.

7.6 BOUNDARY CONDITIONS:

7.6.1 Nozzle Inlet:

Pressure inlet was taken as inlet for nozzle the value of pressure is 8101325 Pascal. Initial gauge pressure was taken as 7898681 Pascal. Temperature was taken as 1583K.

7.6.2 Outlet blades of rotor:

The outlet is set as pressure outlet with a value of 101325 Pascal

7.6.3 Controls set up:

The solution controls are set as listed below: The under relaxation factor was set as given.

Pressure - 0.3
Density – 1
Body forces – 1
Momentum - 0.7
Pressure velocity coupling was taken as Simple

7.6.4 Discretization Equation is selected as given:

Pressure – standard
Density - 1st order upwind
Momentum - 1st order upwind
Turbulence Kinetic Energy (1st order upwind)
Turbulence dissipations rate (1st order upwind)
Energy - 1st order upwind

7.6.5 Initialization:

Solution initialization is done. Initial values of velocity are taken as 186.3 m/s for y direction. Temperature is taken as 1583K

Residual monitorization is done and convergence criteria are set up. The convergence criteria of various parameters are listed below.

Continuity	-	0.001
X Velocity	-	0.001
Y Velocity	-	0.001
Energy	-	0.001

The number of iterations is set up and iterations starts.

8 RESULTS AND DISCUSSION

8 RESULTS FOR NOZZLE:

Nozzle profile which is examined is considered in 2D. As extent of results in normal direction is nothing but 3D flow through the nozzle for given input conditionwith velocity as 183 m/s and maximum output was observed as a 1423 m/s, such that Mach Number is increased from 0.2 to 1.54 in which nozzle is acting as supersonic nozzle andcontours of mach number as shown in figure[8.1]. The velocity contours of nozzle is plotted in figure [8.2], the pressure contours of nozzle is plotted in figure[8.3],and temperature contours of nozzle is plotted in figure[8.4]The velocity, temperature ,mach number, pressure variation along the nozzle is compared with theoretical calculation and with experimental too. These three results are good in agreement with each other.

8.3 RESULTS FOR INTERACTION OF NOZZLE AND ROTOR BLADES:

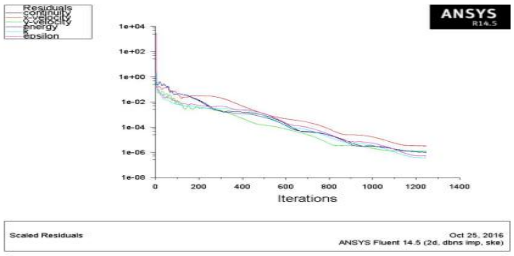
Flow passage form exit of nozzle and entry of turbine rotor blades is allowed under static condition of rotor (blades). As a case of partial admission of axial impulse turbine the flow will suddenly impact to the blades during the of nozzle flow at rotor blades so many factors can be considered to improve the performance of turbine. In this study the tangential velocity is selected as a parameter for better performance of turbine speed. Velocity distribution.

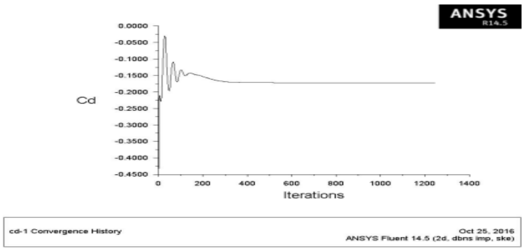
Velocity vector contours are for 3mm,4mm &5mm axial gap respectively By observing the tangential velocity contours for 3mm, 4mm, 5mm gap axial clearances of nozzle and turbine rotor blades the maximum average tangential velocity will act a 3mm axial clearence , 3mm gap of axial clearance will be the betterone.

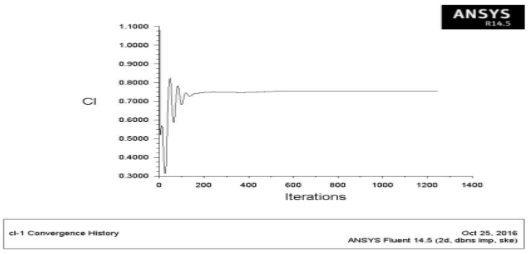
8.4 LIMITATIONS AND SCOPE:

- CD Nozzle flow is carried out for given pressure ration and input output conditions as specified.
- The above nozzle used for this purpose is a special type manufactured by N.S.T.L for their special applications.

- During the study of nozzle and turbine the flow of gases is assumed as at fully length and turbine blades are assumed under staticcondition.







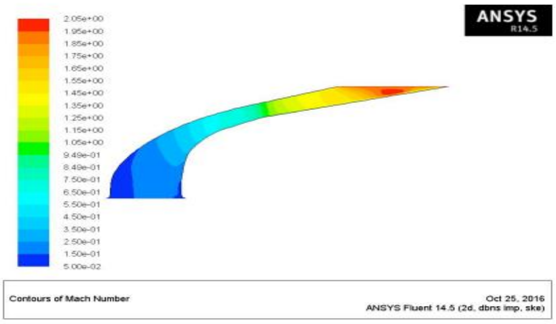


FIGURE : Mach number contours of nozzle.

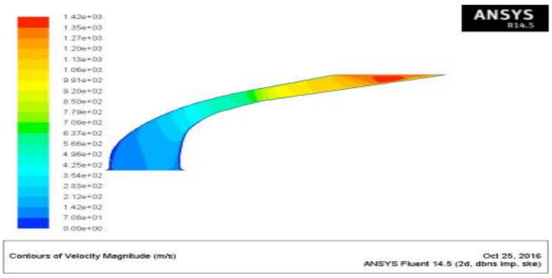


FIGURE : Velocity Contours of Nozzle.

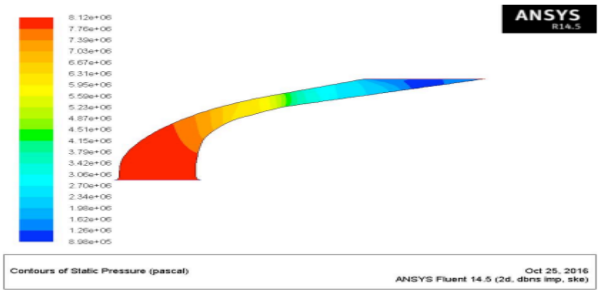


FIGURE : Pressure Contours of Nozzle

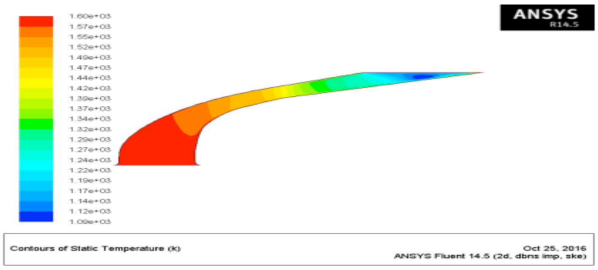
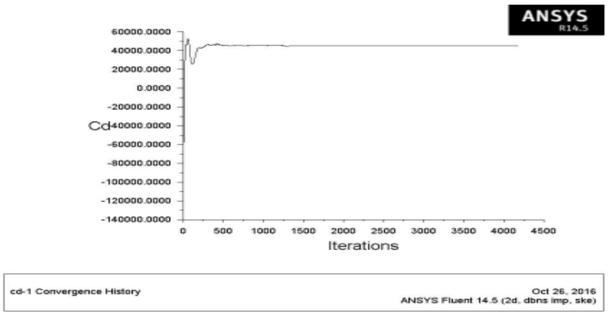
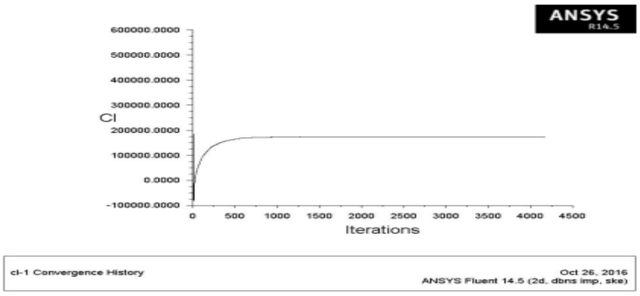
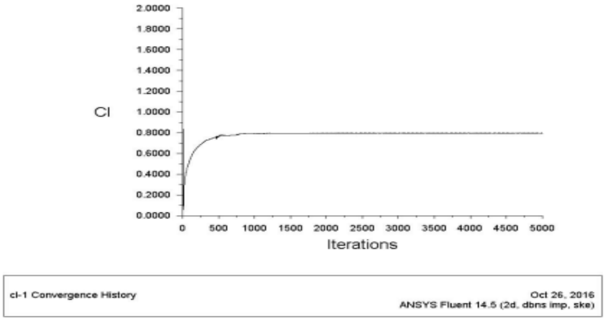


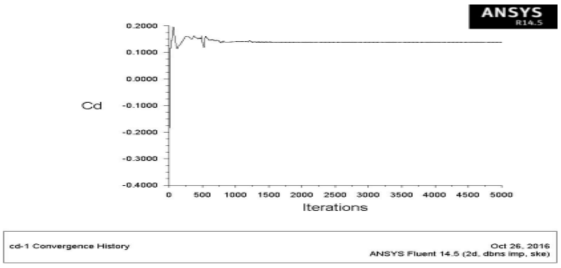
FIGURE : Temperature Contours of Nozzle.







ISSN No: 2348-4845
International Journal & Magazine of Engineering, Technology, Management and Research
A Peer Reviewed Open Access International Journal



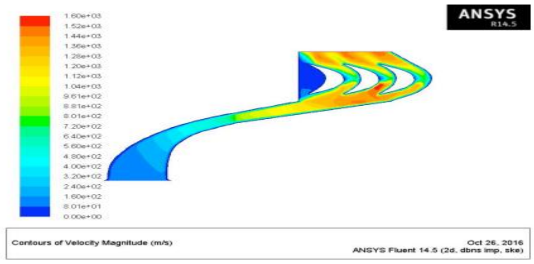


FIGURE: Velocity Contours With 3mm Axial Gap Between Nozzle And Rotor blades.

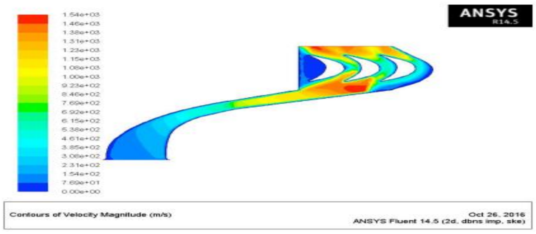


FIGURE : Velocity Contours With 4mm Axial Gap Between Nozzle and rotor blades.

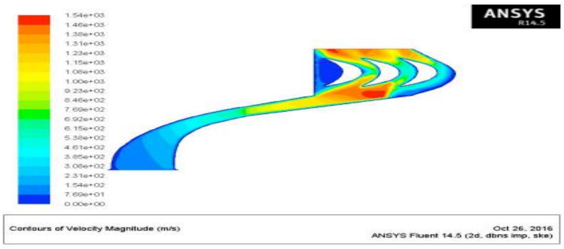


FIGURE : Velocity contours with 5mm axial gap between nozzle and rotor blades.

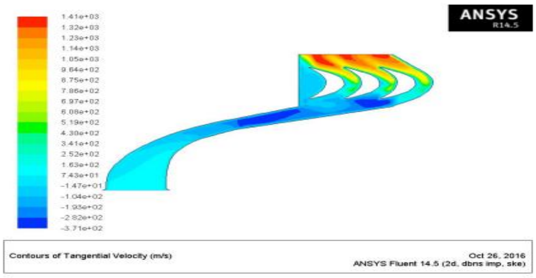


FIGURE : Tangential velocity contours with 3mm Axial gap between Nozzle and Rotor blades.

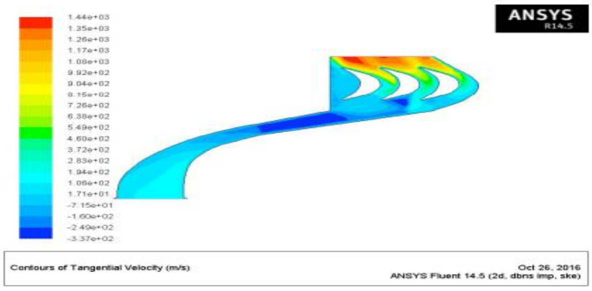


FIGURE : Tangential velocity contours with 4mm Axial gap between Nozzle and Rotor blades

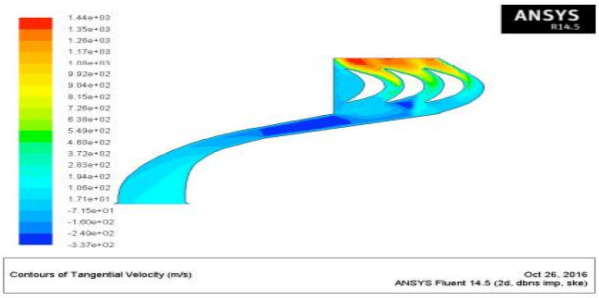


FIGURE : Tangential velocity contours with 5mm Axial gap between Nozzle and Rotor blades.

9.1 CONCLUSION:

9.1.1 FOR THE PRESENT STUDY, THE FOLLOWING CONCLUSIONS ARE DRAWN:

- CFD results of convergent divergent nozzle were in good agreement with the experimental values and theoretical values and the nozzle is acting as a super sonic nozzle.
- CFD predictions for convergent – divergent nozzle and turbine rotor find good agreement with the experimental results of N.S.T.L

9.2 FUTURE WORK:

- Future work should focus on employing a finer resolution grid than the one employed in present study different turbulence models can be tested in the simulations.
- Dynamic mesh can employ for further simulations.
- Nozzle can be studied for different N.P.R to meet the other range of knots.
- Further research can do under dynamic condition of rotor.

REFERENCES

1) Bong-Gunshin, Young-Jae Kwak, Kui-soon Kim and Jin-Han Kim, "A study on nozzle –Rotor Interaction losses for geometric conditions of a partial admission impulse turbine". Proceedings of Asian joint conference on propulsion and power 2006 Beijing, China, April 20-23, 2006.

2) A.A.Khan and T.R.Shem bharkar, "Viscous flow analysis in a convergent – Divergent nozzle". Proceedings of the international conference on Aero space science and Technology, Banglore, India, June 26-28, 2008.

3) H.K.Versteeg and W.Malala Sekhara, "An introduction to Computational fluid Dynamics", British library cataloguing pub, 4th edition, 1996.

4) David C.Wil Cox, "Turbulence modeling for CFD" Second Edition 1998.

5) S. Majumdar and B.N.Rajani, "Grid generation for Abritrary 3-D configuration using a Differential Algebraic Hybrid Method, CTFD Division, NAL, Bangalore, April 1995.

6) Layton, W. Sahin and Volker.J, "A problem solving Approach using Les for a backward facing – step" 2002.

7) M.M.Atha vale and H.Q. Yang, "coupled field thermal structural simulations in Micro Valves and Micro channels" CFD research corporation.

8) Lars Davidson, "An introduction to turbulence Models", Department of thermo and fluid dynamics, Chalmers university of technology, Goteborg, Sweden, November, 2003.

9) Mas Mohas Rai, "Three-Dimensional Navier – Stokes Simulations of turbine Rotor-Stator interaction, NASA Ames Research Center, Moffett Field, California Vol.5, No.3, May – June 1989.

10) Kazuhiro Nakahashi, "Navier – Stokes Computations of two and three dimensional cascade flow fields", Vol.5, No.3, May – June 1989

11) Philip C.E. Jorojenson and Rodrick V.China, " Explicit Range – Kutta Method for unsteady Rotor Stator Interaction", Vol.6, No.6, June 1989.

12) Md.Akhtaruzzaman Sarker and David G.Rhodes, "3D free surface model of laboratory channel with Rectangular Broad – crested weir", proceedings 28th IAHR Congress, Graz, Austria, 1999.

13) Leonid Moroz, Yuri Gororuschenko, Petr Pagur, "Axial turbine stanges design: ID/2D/3D simulation, Experimal, Optimization", Proceedings of GT 2005 ASME Turbo Expo 2005: Nerada USA.

14) Nicholas J. Georgia dis, Teryn W.Dal Bellow, chorles J.Tretry, and Albert L.Johns, "Aerodynamic Design and analysis of High performance nozzles for Mach 4 Accebrator Vehicles", 44th AIAA Aero Space Sciences meeting and exhibit, 9-12 January 2006, Nerada.

15) Eric Gamble and Dan Haid, "Improving off Design nozzle performance using fluid injection" AIAA-2004 – 1206 USA.

16) K.Sukati and Y.Shinizu, "Effect of the increase in the entrance convergent section length of the gun nozzle on the high velocity oxygen fuel and cold spray process", 24th October 2000 ASM international.

17) E.V.Volpe, L.C. decastro sartos "Boundary and internal conditions for adjoins fluid – flow problems". 29 June 2007 Springer science Business Media B.V. 2008.

18) B.Jodoin, "Cold Spray nozzle Mach number limitation', February 14 2001, ASM international.

Author Details

D Chaitan Kumar
M.Tech (Thermal Engineering) Student
Department of Mechanical Engineering
Aditya Institute of Technology and Management, Tekkali

Professor **Dr. N. Haribabu** Was Born Srikakulam, Andhra Pradesh, India. He Has Received P.Hd (Thermal) And M.Tech(MEMH) From ANDHRA UNIVERSITY, VISHAKAPATNAM. HE IS WORKING AS Professor In Department Of Mechanical Engineering, Aditya Institute Of Technology & Management, Tekkali, AP.